

AMES GRANT
IN-34-CR

NASA CONTRACTOR REPORT

127115

Q-10

SOLUTION OF NONLINEAR FLOW EQUATIONS FOR COMPLEX AERODYNAMIC SHAPES

M. Jahed Djomehri

Eloret Institute
3788 Fabian Way
Palo Alto, CA 94303

(NASA-CR-190979) SOLUTION OF
NONLINEAR FLOW EQUATIONS FOR
COMPLEX AERODYNAMIC SHAPES Final
Technical Report, 1 Sep. 1990 - 30
Sep. 1992 (Eloret Corp.) 10 p

N93-12329

Unclass

G3/34 0127115

Prepared for

Ames Research Center
under Cooperative Agreement NCC2-689



National Aeronautics and
Space Administration

Ames Research Center
Moffett Field, California 94035

NASA CONTRACTOR REPORT

SOLUTION OF NONLINEAR FLOW EQUATIONS FOR COMPLEX AERODYNAMIC SHAPES

M. Jahed Djomehri

Cooperative Agreement NCC2-689

CONTRACT NAS2-



SOLUTION OF NONLINEAR FLOW EQUATIONS FOR COMPLEX AERODYNAMIC SHAPES

Final Technical Report

for

Cooperative Agreement NCC2-689

for the period

September 1, 1990 - September 30, 1992

Submitted to

National Aeronautics and Space Administration
Ames Research Center
Moffett Field, California 94035

Larry Erickson, Technical Monitor
Dale R. Satran, Acting Chief, Advanced Aerodynamic Concepts Branch
Leroy L. Presley, Chief, Aerodynamics Division

Prepared by

ELORET INSTITUTE
1178 Maraschino Drive
Sunnyvale, CA 94087
Phone: 408 730-8422 and 415 493-4710
Telefax: 408 730-1441

K. Heinemann, President and Grant Administrator
M. Jahed Djomehri, Principal Investigator

4 November, 1992

Introduction

The design and analysis of future high speed aircrafts requires a realistic consideration of three dimensional flow problems. Some important and interesting flow characteristics about these aircrafts such as shock formation and flow separation from sharp edges are essentially inviscid, rotational physical processes. Thus, the physics of both can be simulated with the Euler equations. The primary obstacle in computing complex flows about realistic aircraft , however, is the difficulty in generating an appropriate grid. Conventional CFD methods based on structured data have difficulty in properly discretizing the entire field of such complex three dimensional geometries in a timely manner; moreover, it is difficult to enhance the accuracy of the solutions near critical regions of the flow in order to provide reliable predictability. Unstructured grids are inherently appropriate for conforming to the complex surface shapes typical of realistic aircraft, and are the key to generating a solution-adaptive grid that can concentrate itself in regions of large flow gradients.

We have been investigating solution-adaptive CFD codes based on unstructured methods for three dimensional complex geometries in subsonic to supersonics regimes and have analyzed the computed solution data in conjunction with experimental data obtained from wind tunnel measurements in order to assess and validate the predictability of the code. We have been particularly involved in the assessment and concurrent improvement of the FELISA code which has emerged as a cooperative agreement between NASA Langley, Ames Research Center and with Swansea and Imperial college in England. FELISA is a package consisting of surface and volume grid generators, Euler flow solvers and solution adaptive grid

programs. This code has been applied to various configurations and has been extensively applied to some generic sonic boom models.

Algorithmic Upgrades:

There are currently two flow algorithms which can solve Euler equations for compressible gasdynamics. One, which is based on a Taylor-Galerkin scheme, has already been discussed in the earlier version of this report. Recently we have examined a new flow solver similar to the previous one, except that the time integration algorithm is based on the multistage Runge-Kutta scheme and is applied to the wing-body example discussed below. It was found that this approach smears out the shock waves more than the Taylor based time integrator.

Many of the CFD applications to practical problems require boundary conditions of special kind. In our cooperative work with Ames-Dryden, for certain engineering problems, multiple boundary conditions have been considered. We have modified the code to handle boundaries such as jet exhaust, mass flow intake/outtake and multiple far field conditions, etc. Moreover, it has been necessary to implement internal boundary conditions for the hybrid methods that is combined structured and unstructured grid approach in order to solve inviscid/viscous flow problems. The issue of concern, such as convergence, accuracy and smoothness of the computed solution in the presence of these boundaries have been studied.

New improved versions of surface and volume grid generators have also been tested on the wing-body example. These versions of grid generators have been found to be more robust than the previous ones and have the capability of grid concentrated about specified points or line segments in the field. Then allow better resolution of high

curvature regions, such as the leading edge, etc., where the related algorithm is based on the Advancing Front Method, a discussion of which has already been presented in the previous reports.

A new program for the solution adaptive scheme based on a new approach in refinement methods has also been tested on the above example. The previously reported adaptivity scheme was based on the remeshing approach where a new grid is regenerated based upon a measure of error of a key variable in the solution data computed on the non-adapted initial grid and automatically generated background grid data. Now, in the new version, one may locally increase the number of grid points in the specified regions by subdividing marked tetrahedral elements of the initial grid. Elements are marked via a measure of solution error of a key variable (similar to the remeshing approach) based on the computation of divided differences over each edge of elements. The addition of new points on the facets of those elements which lie on the surface of the model has also been considered. This has been necessary to assure that the coordinates of the new points conform to the same analytical representation of the surface as used in the initial grid generation procedures. This portion of the program needs to be further modified; in some applications we have encountered misplaced boundary points. Furthermore, it is necessary to include an unrefinement algorithm in order to optimize the total number of elements in the field and avoid the production of an excessively large grid.

Wing-Body Problem:

This has been a third of the generic sonic boom model which has been tested by the FELISA to assess sonic boom signatures. The geometry consists of a slender body with half cone angle about 5 degrees and a double-wedged, diamond-shaped swept back wing with

a step or ramp like connection to sting, at free stream flow condition $M_\infty = 1.68$ and α equal and larger than 0 degrees. The flow characteristics consist of the weak shocks at the apex of the conical body, expansion waves, leading and trailing edge shocks, expansion at the diamond wedge.

Several grids of various size with between 350K elements and up to 1.5 million elements have been generated with various background grids in order to concentrate grid points near the cone apex and the leading and trailing edges. The quality of the grid with regard to skewness of the elements has been studied using different parametric values. Subsequent improvements of the grid generators made in cooperation with our colleagues in the framework of the aforementioned cooperative activities has materialized. Flow solvers based on Taylor Galerkin and Runge-Kutta Galerkin approaches have been tested on selected grids. Here we have studied the accuracy of the solution in comparison with the experimental data. It has been found that the Runge-Kutta approach has been more dissipative in capturing shocks, the results of which have invoked a modification of the numerical diffusions in the code (cooperative work in progress).

Solution adaptive programs have also been used; some of these efforts have substantiated further modification of this program, such as implementation of refinement techniques. Due to an increase in the size of the elements in the field following this approach a prudent course of action which incorporates efficient unrefinement and solution error sensing algorithms must be employed (cooperative work in progress). Computed results of pressure signatures for some grids have been compared with pertinent wind tunnel data measurements. The results are generally in agreement with the experimental data. The pressure signature due to bow shocks at the

apex of the cone is well predicted, the second rise due to the leading-edge shock is slightly underpredicted. This has been found to be related to grid density in the region of space corresponding to Mach-cone, generated by the leading edge; an area which has been difficult to properly concentrate grid points due to some programming problems in the adaptivity code. This problem was cured when additional programming, based on geometric adaptivity in combination with solution adaptivity, was implemented. Although this part has not yet been automated for general cases, it has served the purpose of demonstrating the need for more sophisticated solution-adaptive approaches.

Hybrid Approach:

We have also been involved in developing new techniques based on hybrid approaches, which would allow the solution of Navier-Stokes and Euler flow solvers in conjunction with the structured and unstructured grid approaches. This enables users to benefit from the advantages of structured grids to capture those characteristics of the flows, such as boundary layers, viscous effects, turbulence, etc., which are normally too complicated to be handled by unstructured methods. At the same time, however, issues relating to the complexity of the geometries are handled by the unstructured approach. These activities have been launched cooperatively with other researchers. We have developed a pilot code and a test problem to assess the qualitative and quantitative aspects of the approach and to study the interface boundary conditions which are presented by this approach. Work is underway on a test problem consisting of a of conical body of revolution at a high angle of attack in subsonic regions. We have, sofar, been successful in smooth transmission of solution data across boundaries between viscous regions governed by the Navier-Stokes and the inviscid regions governed by the Euler

equations. Convergence of the combined solution retaining the vortical structures of the primary and secondary vortices in the field has been quite successful as compared with just the structured solution using the Navier-Stokes equations over the entire field.

Cooperative Works and Consultations:

We have been continuously consulting and supported Dr. K. Gupta and his colleagues on CFD issues related to engineering problems at the Ames-Dryden research facilities working toward his multidisciplinary program to combine flow codes with structural dynamics codes. We have incorporated multiple boundary conditions capability to the FELISA code and modeled problems of interest, such as the simulation of pegasus configurations, the pressurized cabin problem, a model of rotor-aircraft nacell-jet-exahusts-deflector problems, and a model of the burner-body configurations of SR71, etc.

We have also consulted other users At Ames and industries such as Boeing Helicopter and Lockheed on the FELISA code. Other related works involved activities related to the assessment of graphics and visualization codes such as Wave fronts, Visual3 (from MIT), the inhouse developed graphics package, known as Giraffe. We have tested and set examples and modified input/output issues related to these softwares.